The use of upwind schemes at high Reynolds number—a cautionary note

W.Arter

CLM-P835

The use of upwind schemes at high Reynolds number—a cautionary note

Wayne Arter

Culham Laboratory, Abingdon, Oxfordshire, OX14 3DB, U.K.

(Submitted for publication in Journal of Computational Physics).

January 1988

Simple upwind schemes are the basis of many codes of practical importance for design studies in heat transfer and fluid flow applications, e.g. the FLOW3D code [1]. With care, it is possible to produce codes that can give bounded answers regardless of mesh-spacing or time-step size. This note highlights a rather disturbing inaccuracy that can arise if the mesh-spacing ~~Δx~~ is taken too large.

The relevant dimensionless measure of Δx for fluid flow calculations is the mesh Reynolds number R_m, defined like the usual Reynolds number R, but with Δx as length-scale. It has become customary [2,3] to study the effects of large R_m using the viscous Burgers' equation as a model. Following the work of Cheng and Shubin [2], hereinafter referred to as "CS", we consider steady state flows $u(x)$ which must satisfy

$$R\,u\,\frac{\partial u}{\partial x} = \frac{\partial^2 u}{\partial x^2} \qquad (1)$$

subject to boundary conditions $u(\pm 1) = \mp 1$.

The boundary conditions of oppositely directed flow lead at large R to the formation of a viscous layer at $x = 0$ where the flow u abruptly changes direction. The problem is sufficiently simple that an analytic solution is available, namely

$$u(x) = -\,\alpha \tanh(\alpha R x/2), \qquad (2)$$

where α is the positive root of

$$\alpha \tanh(\alpha R/2) = 1\ . \qquad (3)$$

Evidently the viscous layer has thickness $1/R$; when this is small relative to the mesh-spacing employed in a numerical calculation, we shall see that troublesome errors may occur.

For the purposes of this note, the antisymmetry about $x = 0$ means we need only consider what happens on the interval $x = 0$ to $x = 1$. We introduce a uniformly spaced mesh, with points

$$x_i = i\,\Delta x, \qquad i = 0, \ldots, N, \qquad (N\Delta x = 1), \tag{4}$$

and seek values u_i, $i = 0, \ldots, N$, to approximate $u(x_i)$. The operators $u\partial u/\partial x$ and $\partial^2 u/\partial x^2$ must also be discretised.

For $u\partial u/\partial x$ we use the customary conservative upwind difference scheme as proposed by Patankar [4] for advection of a scalar, such as temperature T. In that situation it is convenient to use staggered meshes, i.e. u_i and T_i are defined at points separated by $\Delta x/2$. If this formalism is applied to Burgers' equation, it follows we need to have some way of approximating u at points $x_{i+1/2} = (i+1/2)\Delta x$. (A slightly different treatment is required to give the conservative upwind scheme described in CS.) Inevitably we take

$$u_{i+1/2} = (u_i + u_{i+1})/2, \tag{5}$$

giving (since $u_i < 0$ for $i > 0$) the discrete analogue of Burgers' equation

$$R\frac{\left[\frac{1}{2}(u_i + u_{i+1})u_{i+1} - \frac{1}{2}(u_{i-1} + u_i)u_i\right]}{2\,\Delta x} = \frac{u_{i-1} - 2u_i + u_{i+1}}{(\Delta x)^2}, \qquad i = 1, \ldots, N-1. \tag{6}$$

(We have taken the customary 3 point centred difference formula for the diffusion operator.) Eq.(6) can be used to formulate the CS problem as

$$u_{i+1}^2 - u_i^2 + u_i(u_{i+1} - u_{i-1}) = \frac{4}{R_m}(u_{i-1} - 2u_i + u_{i+1}),$$
$$i = 1, \ \ldots, \ N-1, \tag{7}$$

subject to $u_0 = 0$, $u_N = -1$.

Following CS, we sum over equations (7); because the scheme is conservative, internal contributions cancel pairwise, giving

$$-u_1^2 - u_o u_1 + u_N^2 + u_{N-1}u_N = \frac{4}{R_m}(u_o - u_1 - u_{N-1} + u_N). \tag{8}$$

Since u_0 and u_N are given, this is a quadratic equation for u_1 in terms of u_{N-1}. Writing $u_{N-1} = -1 + \varepsilon$, we obtain

$$u_1 = \frac{2}{R_m} \pm \surd\{(\frac{2}{R_m})^2 + 2 - \varepsilon - \frac{4\varepsilon}{R_m}\}. \tag{9}$$

By Taylor expanding the exact solution (2) about $x = 1$, we find

$$\varepsilon \simeq \frac{R_m}{2\sinh^2(\alpha R/2)}. \tag{10}$$

For $R >> 1$, (3) implies $\alpha \simeq 1$, hence both ε and ε/R_m are small, and

$$u_1 \simeq \frac{2}{R_m} \pm \surd\{(\frac{2}{R_m})^2 + 2\}, \tag{11}$$

i.e. as $R_m \to \infty$, $u_1 \simeq -\sqrt{2}$ (we take the negative root because (7) assumes $u_i < 0$, all i).

The exact solution (2) is monotone and for large R, $u \simeq -1$ everywhere outside a narrow layer. Hence the scheme (6) produces a result in error by $\simeq 40\%$ if R_m is too large, or equivalently $\Delta x \gtrsim 1/R$, i.e. the viscous layer is not resolved. The worrying feature is that the misleading result is obtained for any sufficiently high R_m, thus halving Δx need not change the value of the velocity at the first interior mesh-point. Only by checking the spatial distribution of the computed u_i can the error be detected, see Fig. 1.

There remains the question of error behaviour at smaller R_m. Fig. 2 plots the difference E between u_1 (eq.(11)) and the true value of $u(\Delta x)$ from (2). We see that the error E passes through zero for $R_m \simeq 3.3$, and thus that $E < 3\%$ provided $R_m < 3.8$. Over this range of R_m, the Patankar scheme outperforms the conservative upwind scheme in CS which gives errors of up to about 17%. Misleading results are obtained only at higher R_m.

Lest it be thought that the model problem is too idealised to be of practical use, we remark that a 40% error in velocity will lead to a similar error in heat fluxes, other things being equal. A mesh refinement study was conducted, using the FLOW3D code, for the Rayleigh-Benard problem in a closed box of aspect ratio 3/2 (see [5] for more details). The results obtained, for a Rayleigh number Ra = 120000 and Prandtl number Pr = 2.5 are listed in Table I. Observe that the error in heat flux Nu depends on R_m much as we should expect from Fig.2, and that the error is as large as 25% when $R_m \simeq 7$.

Detailed inspection of the flow field from the finer mesh reveals the presence of small counter-circulating "Moffatt" eddies in the corners of the box. Overall, flow is more vigorous on the finest mesh, R = 49

against $R = 40$ on the coarsest, but in a corner, at e.g $(x,z) = (17/12, 1/6)$ we have that the vertical component of velocity $U_z = -12$ as against $U_z = -22$ on the coarser mesh with $\Delta x = 1/6$. The greater speeds in all four corners are apparently ultimately responsible for the increased heat flux Nu, since the breakdown of Nu into conductive and convective contributions (at $z = 1/24$) is: 4.01 and 0.96 on this coarse mesh, 3.84 and 0.12 on the finest mesh. Obviously the situation is not as clear cut as could be desired. Studies even at low Prandtl number show that the linkage between velocity and temperature fields persists (i.e. we cannot find an accurate temperature distribution unless the velocity is correspondingly well represented). However, a two-dimensional version of the above, spurious velocity enhancement mechanism, must be a strong candidate to explain the faster flow on the coarser meshes.

Acknowledgements

The author thanks the Harwell group for making available the FLOW3D code, and for valuable discussions.

References

1. I. P. JONES, J. R. KIGHTLEY, C. P. THOMPSON and N. S. WILKES, 1985 "FLOW3D, a computer code for the prediction of laminar and turbulent flow, and heat transfer: Release 1", Harwell report AERE-R11825 (1985).

2. S.I. CHENG and G. SHUBIN, J. Computational Phys. 28, 315(1978).

3. J. W. EASTWOOD and W. ARTER, IMA J. Num. Anal. 7, 205 (1987).

4. S. V. PATANKAR, "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York (1980).

5. W.ARTER, A. BERNOFF and A.C. NEWELL, Phys. Fluids 30,3840 (1987).

Table I

Mesh refinement study using FLOW3D for the Rayleigh-Benard problem in a closed box of aspect ratio 3/2 at a Rayleigh number of 120 000 (Prandtl number 2.5). R_m is mesh Reynolds number and Nu is the total heat flux, across a plane near a horizontal boundary, normalised with respect to the conductive flux in the absence of convection.

$1/\Delta x$	R_m	Nu
4	10.	4.541
6	7.2	4.967
12	3.7	3.940
24	2.0	3.951
48	1.0	3.963

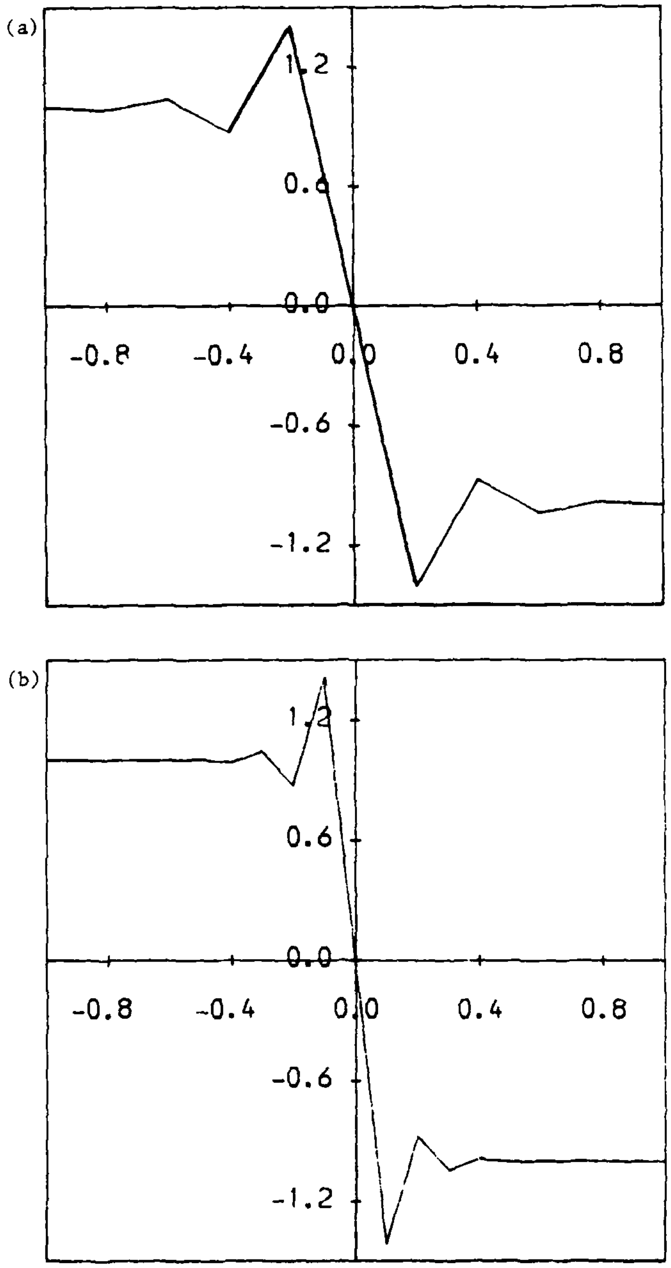

Fig. 1 Steady state solutions at $R=500$ for the viscous Burgers' equation, obtained using the Patankar scheme with (a) $\Delta x=0.2$, (b) $\Delta x=0.1$. Observe that reducing R_m by a factor of two has no effect on the maximum.

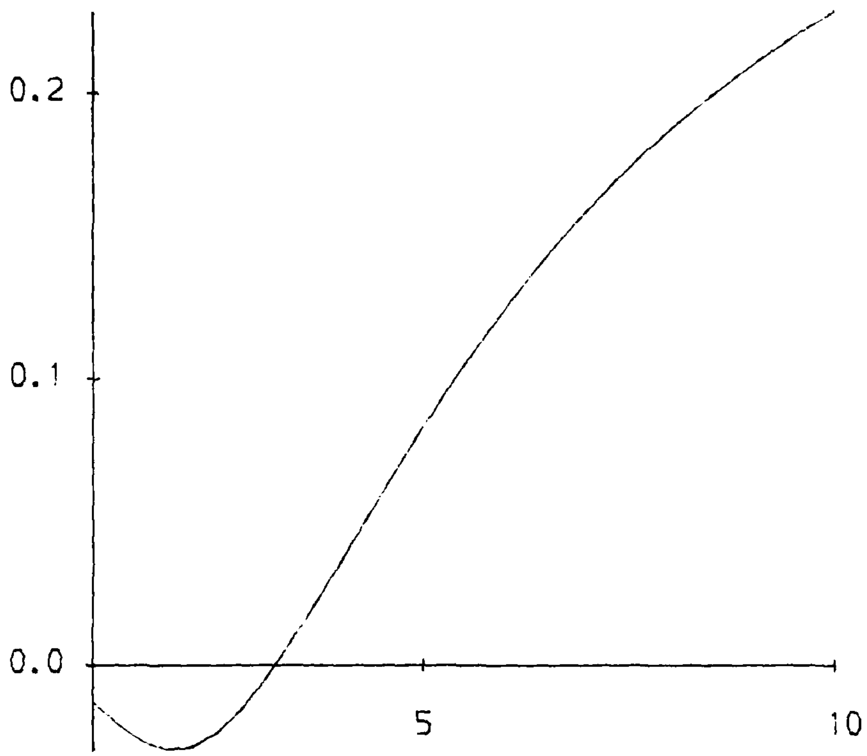

Fig. 2 E is the error in the solution to the viscous Burgers' equation obtained using the Patankar scheme. E is plotted against R_m for $1 \leq R_m \leq 10$. (For this plot $\Delta x = 0.01$, but other plots with $\Delta x \leq 0.1$ are almost identical).